CAD Toolkit User Manual

LUSAS Version 14.5 : Issue 1

LUSAS
Forge House, 66 High Street, Kingston upon Thames,
Surrey, KT1 1HN, United Kingdom

Tel: +44 (0)20 8541 1999
Fax +44 (0)20 8549 9399
Email: info@lusas.com
http://www.lusas.com

Distributors Worldwide

Copyright ©1982-2010 LUSAS
All Rights Reserved.

Table of Contents

Introduction 1
- **LUSAS CAD Toolkit** 1

Modeller Import and Export 1
- **DXF Import and Export** 1
- **IGES Import** 1
- **CADA-X Export** 1
- **PATRAN Neutral File Import and Export** 2
- **PATRAN Interface** 2
- **FEMView and FAM Export** 2

PATRAN Interface 3
- **Introduction** 3
- **Installation** 3
- **PATRAN to LUSAS (PATLUS)** 4
- **LUSAS to PATRAN (LUSPAT)** 10
- **Interface Integration with PATRAN** 14
- **PATLUS Element Details** 19
- **LUSPAT Results Columns** 27
- **System Parameters** 30

FEMView and FAM Interface 33
- **Introduction** 33
- **LUSAS to FEMView/FAM** 33
- **FEMView/FAM Stress Types** 37

Introduction

LUSAS CAD Toolkit

This manual covers interfaces to the LUSAS programs involving the use of external pre- and post-processing packages. The following interfaces are provided for use with LUSAS:

- **DXF Import and Export**
- **IGES Import and Export**
- **LMS CADA-X Export**
- **STEP Import and Export**
- **STL Import and Export**
- **PATRAN Neutral File Import & Export**
- **PATRAN Interface (PATLUS, LUSPAT)**
- **FEMView and FAM Interface (FVLUSAS)**

DXF, IGES, LMS CADA-X, STEP, STL and PATRAN Neutral File import/export facilities are provided in LUSAS Modeller. The use of PATRAN, FEMVIEW and FAM interfaces are described in this manual.

Modeller Import and Export Capability

DXF Import and Export

DXF is a file format initially developed for import and export of drawing data to and from AUTOCAD. Many programs now support the DXF format and it is often used to transfer geometry data between a variety of system. LUSAS Modeller supports both the import and export of DXF data. Full details can be found in the *Modeller Reference Manual*.

IGES Import and Export

IGES is an international standard format for the import and export of Geometry data. Many CAD/Modelling systems support the IGES format. LUSAS Modeller supports the import and export of IGES data. Full details can be found in the *Modeller Reference Manual.*

CADA-X Export

CADA-X is a format developed by LMS International to transfer data to and from the LMS LINK modal correlation software package. Full details can be found in the *Modeller Reference Manual.*

STEP Import

STandard for the Exchange of Product data (STEP) files are imported according to Part 42 of the Geometric and Topological Representation by using the File > Import menu item. When a file is selected the import process may be controlled from the Advanced button by specify the parameters.

LUSAS Model geometry cannot currently be exported to a STEP file.

STL Files

STL files are used by Stereolithography software. They hold information needed to produce 3D models on Stereolithography machines. STL files are imported from the File> Import menu item. When a file is selected the import process may be controlled from the Advanced button by specify the parameters. LUSAS Model geometry may be exported to IGES from the File> Export menu item.

PATRAN Neutral File Import and Export

The PATRAN Neutral File format was originally developed for the import and export of data to and form PATRAN. As well as the PATRAN interface to the LUSAS Solver, LUSAS Modeller supports the import of PATRAN Geometry (Phase 1) and the export of finite element data (Phase 2). Full details can be found in the *Modeller Reference Manual*.

PATRAN Interface

The LUSAS-PATRAN Interface transfers data between LUSAS Solver and PATRAN. PATRAN is used to generate a neutral file which contains all of the information required to generate a valid LUSAS data deck. The conversion of the data is achieved using the program PATLUS.

The conversion of results from LUSAS format to PATRAN format is handled by the program LUSPAT. LUSPAT takes a LUSAS restart file and generates PATRAN results files for each load case and results type.

FEMView and FAM Export

FVLUSAS is the LUSAS Solver to FEMView/FAM export program supported by FEA Ltd.

The program reads results from the LUSAS restart file. The LUSAS restart file is named **jobname.rst.** A restart file is created by LUSAS using the RESTART WRITE or RESTART WRITE BRIEF commands in the data file or using the appropriate OUTPUT option in various analysis control sections in the data file.

PATRAN Interface

Introduction

This section describes the LUSAS-PATRAN Interface for the transfer of data between LUSAS Solver and PATRAN.

PATRAN is a registered trademark of the MSc Corporation.

Installation

All software required for your LUSAS to PATRAN interface should be installed automatically with any software update from FEA Ltd. The files required for running the LUSAS to PATRAN interface (listed below) are all supplied on the standard LUSAS Release Kit.

The following files will be installed in the **$FEADIR** directory:

patlus	Script for running the PATRAN to LUSAS translator.
patlus.exe	PATRAN to LUSAS translator.
luspat	Script for running the LUSAS to PATRAN translator.
luspat.exe	LUSAS to PATRAN translator.

The following files will be installed in the **$FEADIR/config** directory:

patlus.str	Optional start-up file for the PATRAN to LUSAS translator.
luspat.str	Optional start-up file for the LUSAS to PATRAN translator.

The following files will be installed in a directory called **$FEADIR/patran3** and its sub-directories.

lusas.db	Database template for the LUSAS preference.
lusas.plb	PCL library for the LUSAS preference.

lusas_dis.res_tmpl	Results template file.
lusas_nod_els_2d_thermal.res_tmpl	Results template file.

lusas_nod_els_3d_thermal.res_tmpl	Results template file.
lusas_nod_els_membrane_stress.res_tmpl	Results template file.
lusas_nod_els_plane_strain.res_tmpl	Results template file.
lusas_nod_els_plane_stress.res_tmpl	Results template file.
lusas_nod_els_shells_stress.res_tmpl	Results template file.
lusas_nod_els_solid_axisymmetric.res_tmpl	Results template file.
lusas_nod_els_solid_stress.res_tmpl	Results template file.
pat3_lusas.ses	Forward translator test session file.
lusas_pat3.ses	Reverse translator test session file.

If it is required to run PATLUS/LUSAS/LUSPAT from within PATRAN, the library of supplied files will need to be installed as root using the following procedure:

Step One	Go to the **patran3** subdirectory in your main LUSAS release area containing the required integration tools, for example: `cd $FEADIR/patran3`
Step Two	Copy the necessary integration tool files into the **$P3_HOME** directory and its associated sub-directories **res_templates** and **test_files**, by executing the installation script by typing: `./p3install`

PATRAN to LUSAS (PATLUS)

The conversion of PATRAN data to LUSAS data is achieved using the PATRAN neutral file system. PATRAN is used to generate a neutral file which contains all of the information required to generate a valid LUSAS data deck. The conversion of the data is achieved using the program PATLUS. The input to PATLUS is the neutral file **filename.def** which is then translated to the LUSAS data deck **filename.dat**. PATLUS is invoked by typing **patlus** and specifying the root of the filename to be converted:

```
patlus filename
```

Alternatively, PATLUS and LUSAS may be run directly from within PATRAN using the scripts provided and detailed in this manual.

The elements and loading types supported are shown on page *19*.

LUSAS Data Deck

The LUSAS data deck generated by PATLUS will contain some or all of the following data sections:

Problem Title	This is read from the PATRAN neutral file.
Options	OPTIONS 2 117 118 are inserted automatically. Any further options should be input manually.
Element Topology	The currently available elements are listed on page *19* together with their PATRAN identification and configuration numbers. For 2D models all element normals should be consistent throughout the structure. The PATRAN user should ensure that all element local axes are consistent, throughout the structure, with the (Hyper) Patch local axes. See the notes on beam and joint orientation at the end of this table.
Solution Order Automatic	This data chapter is inserted automatically. The Sloan optimiser is selected by default.
Node Coordinates	This data chapter is generated automatically. PATLUS will automatically insert additional dummy nodes for the orientation of beams, joints and coordinate frames.
Geometric Properties	This data chapter is inserted automatically.
Geometric Assignments	This data chapter is inserted automatically.
Material Properties	Isotropic Material Properties are available. Composite properties are not supported and should be input manually.
Material Assignments	This data chapter is inserted automatically.

Slideline Properties	This data chapter is inserted when slidelines are generated. All slidelines are initially assigned the default properties as defined in the *Modeller Reference Manual*.
Slideline_Surface Definition	This data chapter is inserted when slideline data is generated within PATRAN using pressure loading with reserved data values. Slidelines are assigned by applying pressures. A load case should be used exclusively for slidelines. **Method 1 (INTSL3=0)** Master slidelines are assigned a loadset-id in the range 1000-1999 and the corresponding slave slideline is assigned a loadset-id in the range 2000 - 2999 by adding 1000 to the master slideline loadset-id. The pressure value is ignored. **Method 2 (INTSL3=1)** Slidelines are assigned by inserting a code in the element pressure field. The magnitude of the first component of the applied pressure applied within this load case represents a code: `pressure = nnnst` where: **nnn** indicates slideline number. The number must be greater than 99). **s** indicates whether it is a master (1) or a slave (2) slideline. **t** indicates the slideline contact type. Options are: (1) sliding, (2) Sliding with friction, (3) Tied slidelines, (4) Sliding only with no lift-off. For example, slideline 111 allowing sliding with friction is defined by pressure load datasets of magnitude 11112 (master) and 11122 (slave).
Slideline Assignments	This data chapter is inserted when slidelines are generated. All slidelines are assigned property set 1.

Cartesian Sets	This data chapter is inserted automatically. **Note.** Cartesian and cylindrical sets are supported but spherical sets are not currently supported in LUSAS.
Transformed Freedoms	This data chapter is inserted automatically. **Note.** If a CID is assigned to a node in PATRAN the TRANSFORMED FREEDOMS will be used to reflect the transformation in LUSAS. This information is stored in the LUSAS database allowing the user to generate stress and displacement nodal results in either global or transformed directions.
Constraint Equations	This data chapter is inserted automatically when multi-point constraints (MPCs) are specified in PATRAN.
Support Nodes	This data chapter is inserted automatically. **Notes** • Each PATRAN data file must contain only one support case. • Non-standard nodal freedoms are translated in order, for example, a fully restrained semi-loof shell will restrain freedoms 1 to 5, whereas a fully restrained BMS3 will restrain freedoms 1 to 6.

Load Case	The currently available loading types for each element are listed on page *19*. **Notes** • PATRAN does not allow constant body forces to be applied. This data chapter must, therefore, be input manually. • When non-standard freedoms are loaded, the loading should be entered in the order the freedoms appear in LUSAS, e.g. the semiloof elements require loof moments M1 and M2. These should be input in positions Mx and My. • For prescribed displacement loading the PATRAN constraint set identification number must be used to assign this loading type to the required load case. • For field elements the face load data input should be input to PATRAN in the C1 flag direction.
Analysis Control	Some analysis control options are available within PATRAN. See pages *8* and *16* for more information.

Element Output Control	All element output is suppressed by default. If specific output is required after the LUSAS run has completed this can be obtained using the LUSAS restart facility. For example:
	`RESTART READ 1`
	`LOAD COMBINATION`
	`1 1`
	`OPTION 55`
	`ELEMENT OUTPUT CONTROL`
	`1 10 1 1 1`
	`NODE OUTPUT CONTROL`
	`1 20 1 3`
	`END`

Node Output Control	Only reactions are output by default. Displacements are suppressed.
Restart Write Brief	This LUSAS command is output automatically.
End	This LUSAS command is output automatically.

Notes on Beam/Joint Orientation in PATRAN

There are two ways of orienting line elements in PATRAN:

- **Node** Using a node to define the xy-plane.
- **Local Coordinate System** Using a local coordinate system to define the local x axis and xy plane.

These are used with two different types of elements:

- **Beam elements** These have a finite length and hence a node is sufficient to define the beam axes (x is taken as along the beam axis). All LUSAS beams in PATRAN are oriented using a method 1 above, but the node is an optional parameter. The usual rules apply regarding defaults.
- **Joints** These are normally zero length, and hence require a local coordinate system to define both the local x axis and the xy plane. All LUSAS Joint

elements in PATRAN are oriented using method 2 above, but the local coordinate is an optional parameter. The usual rules apply regarding defaults.

Element local axes are displayed from within the **Element Properties** dialog box. Use the following procedure:

1. Set the Action to **Show**. All of the element properties defined will be displayed.
2. Choose **X-Y Plane Orient. Node** for beams (see note 3 below), or **Orientation System** for joints (see note 4 below).
3. For beams, the following selections are available:
   - **Table** produces a report table.
   - **Marker Plot** marks the elements with their orientation node numbers.
   - **Scalar Plot** produces a fringe plot of orientation node numbers.
4. For joints, the following selections are available:
   - **Table** produces a report table.
   - **Vector Plot** marks the elements with a view of the local coordinate used to orient the joint, provided that the elements have been added to the appropriate Group and the **Functional Assignments Display** setting has been toggled to **Display on FEM Only**.

LUSAS to PATRAN (LUSPAT)

The conversion of results from LUSAS format to PATRAN format is handled by the program LUSPAT. LUSPAT takes a LUSAS restart file and generates PATRAN results files for each load case and results type.

Since the results are created from the LUSAS restart file this should be first created by inserting RESTART WRITE BRIEF in the LUSAS data set. This line is written automatically by PATLUS.

Interactive Use

If LUSPAT is to be used in an interactive mode the user will be prompted for the following information.

```
File:
```

The program will initially prompt the user for the input job file. Input the LUSAS filename without the **rst** extension.

```
Enter model name (<CR>for jobname)_:
```

An optional model name is prompted for which is used to generate the results file names. If omitted the **jobname** is used. When run in batch mode the model name will default to the **jobname**.

Enter RESTART DUMP number required (zero for all dumps)_:

The program will prompt for the RESTART DUMP number. For linear and eigenvalue extraction problems, dump number 1 should be specified. For nonlinear or transient problems all the dumps may be translated by specifying zero.

Do you require eigen analysis + nonlinear results [N]_:

If an eigenvalue analysis has been preceded by a nonlinear analysis the program will inquire whether the eigen analysis plus the nonlinear results are to be output.

Enter LOAD CASE/MODE required (zero for all loads)_:

For linear or eigen problems the load case/mode number must be input. If a zero is specified all load cases will be translated. If a load case is specified the prompt will be repeated until a carriage return is specified. When more than one load case/mode is specified the results will be combined with the corresponding load factors. Up to 50 load combinations can be processed at any time.

Enter FACTOR required

For each load case/mode specified a corresponding factor must be input. If all load cases are to be translated a factor of one is used.

Enter STRUCTURE TYPE	
Beam	**-1**
Plane Membrane	**-2**
Space Membrane	**-3**
Plate Flexure	**-4**
Shell	**-5**
Axisymmetric/Plane strain	**-6**
Solid	**-7**
Field 2D	**-8**
Field 3D	**-9**
Field 1D	**-10**
Enter number _:	

To prevent invalid data being assembled at common nodes, i.e. between beams and shells, the structure type under consideration must be input. If a model contains two or more structure types the translation process should be repeated for each structure type required. If zero is specified auto structure type selection will be used. When several

structure types are present the structure type selected will be the first identified in the results file.

```
Enter material number required (zero for all
materials)_:
```

Enter the selected material number as defined in the LUSAS material assignments to enable averaging of stresses for that material only. This feature allows selective nodal averaging of different parts of the structure and may be used, for example, to carry out selective nodal averaging of a branched shell. In this case each branch of the shell should be modelled in PATRAN with a different material assignment.

```
Do you require nodal results in transformed directions
[N]_:
```

The displacements and stresses may be output in the transformed directions as defined in the LUSAS transformed freedoms data chapter if required. The transformed freedom assignments in LUSAS are controlled by the CID of the nodes in PATRAN.

```
Do you wish to continue [N]_:
```

Further load cases or modes can be processed by continuing. The default answer stops the program.

Batch Mode

LUSPAT has been designed to be interactive but may be run in batch by setting the system parameter IBATCH=1 in the LUSPAT start-up file **luspat.str** as described on page *30*. The prompts will then take the following default values which may be overridden as indicated:

- All restart dumps will be translated, unless the system parameter **NDUMP** is set to the required dump number.
- All load cases will be translated.
- Auto structure type selection will be used, unless the system parameter **NSTRUT** is set to the required structure type number.
- All material types will be translated together, unless the system parameter **NMAT** is set to the required material type number.
- Eigenvector results will not be superimposed on the nonlinear deformation unless the system parameter **IEIGNL** is specified.
- Results will be translated in the global coordinate system, unless the system parameter **IRSTRF** is specified.

Full details of supported system parameters are given on page *30*.

File Naming Convention

The conversion used for the result file names produced is:

```
model_number.ext
```

where **model** is the specified model name, **number** is the LUSAS restart dump number or the LUSAS load case or eigen mode number, and **ext** is the extension which indicates the file content. The extensions adopted are:

name.def	structure definition
model_number.adp	nodal displacements (.dsp for binary file)
model_number.and	nodal results (.ndl for binary file)
model_number.ael	element results (.elm for binary file)

where:

name is the **jobname** corresponding to the restart file name.

For example, **jobname** for the file **test.rst** is **test**.

Note. On certain machine types the documented file naming convention is not adhered to due to system limitations.

Results File Titles

All results file titles are automatically assembled to include the problem title, the selected material number and the structure type. For nonlinear analysis the load increment and arc length load factor are output. For transient analysis the time step and response time are output. For frequency analysis the mode number, eigenvalue, frequency and error norm are output. For buckling analysis the mode number and load factor are output.

The titles given in the displacement, nodal results and element results files take the form:

```
TITLE:  problem title
TITLE1: model_number structure_type results_type
material_no
TITLE2: analysis data
```

where:

problem title	is as defined in the LUSAS data file
model_number	is as defined in the file name, i.e. test_001
structure_type	is as selected, i.e. shell

results_type is the description of the results, e.g. TRANSFORMED AVERAGED RESULTS
material_no is the material group selected
analysis data is defined differently for each problem type. Options are: Linear (blank), Frequency (frequency, error norm), Buckling (load factor), Nonlinear (load increment, load factor), Transient (time step number, response time).

Interface Integration with PATRAN

Introduction

LUSAS has been integrated within PATRAN 3 and above using PCL This enables LUSAS to be run directly from within PATRAN using a single command or menu pick. From within PATRAN the user can use the LUSAS defined menus to set LUSAS up to run different analysis types, control solution optimisation, set output file options and set up the results types and files to be read.

The integrated program uses the following files (which should be installed in the directory **$P3_HOME**):

lusas.plb LUSAS PCL Library of menu functions.
lusas.db LUSAS database template.

The LUSAS preference modifies the following main areas of PATRAN:

- **Materials** Material properties are entered which generate the equivalent LUSAS MATERIAL PROPERTIES data chapters.
- **Element Library** Elements are selected from those available in the LUSAS element library. See page *19*.
- **Loads and Boundary Conditions** LUSAS-style Loads, Supports and Constraint Equations can be entered.
- **Analysis Forms** The analysis dialogs are configured to suit a LUSAS analysis, with options available for controlling the LUSAS solution procedure.

Using Interfaces with PATRAN

The analysis and post-processing procedure may be controlled fully using the **Analysis** dialog box within PATRAN.

PATRAN to LUSAS

Data is transferred from PATRAN to LUSAS by setting the **Action** on the **Analysis** dialog to **Analyze**. The following **Action** settings are available:

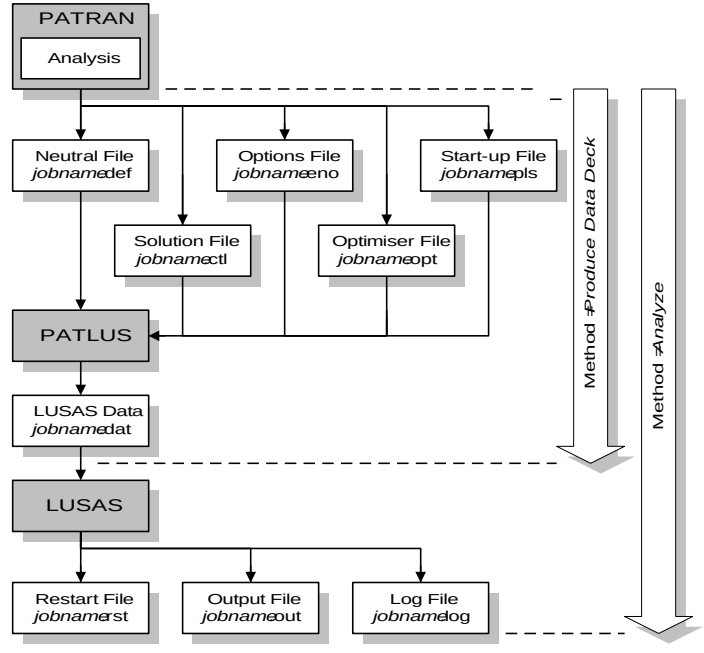
- **Run Analysis** translates the data and carries out an analysis.
- **Produce Data Deck** translates the data to a LUSAS data deck, but no analysis is carried out.

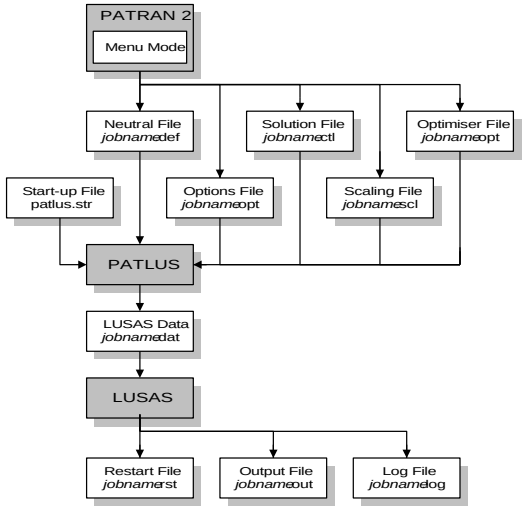
The type of analysis to be carried out is selected using the **Solution Type** dialog. This will generate a file **jobname.ctl** which is picked up by PATLUS. The default solution type is linear static.

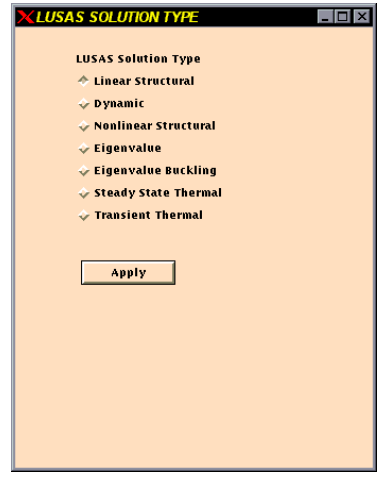
The type of frontwidth optimiser to be used during solution is selected using the **Optimiser Type** dialog. This will generate a file **jobname.opt** which is picked up by PATLUS. The default optimiser type is Sloan.

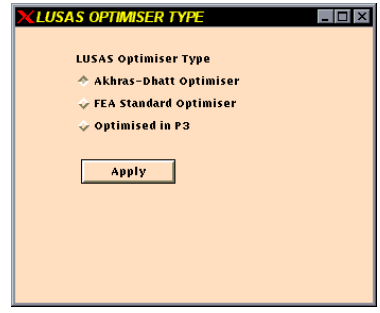
The amount of output written to the LUSAS text output file is controlled using the **Output Requests** dialog. This will generate a file **jobname.eno** which is picked up by PATLUS. The default is for output of reactions at nodes only. This does not affect the results available for viewing in PATRAN which are extracted from the restart file.

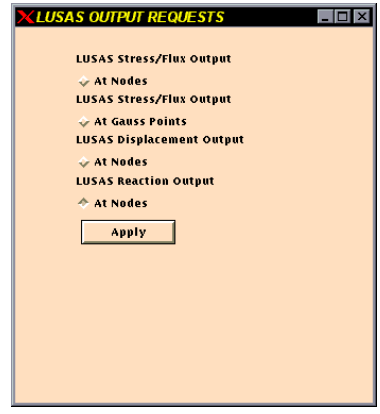
PATRAN to LUSAS Translation Procedure











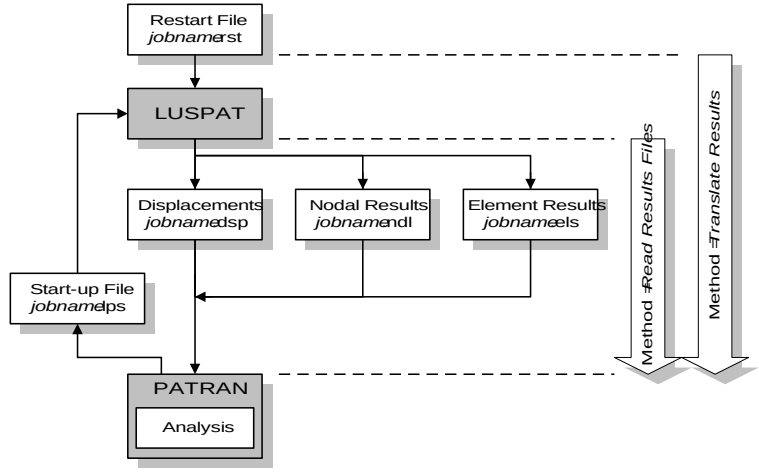
Analysis Dialog for PATRAN to LUSAS Subordinate dialogs shown right.

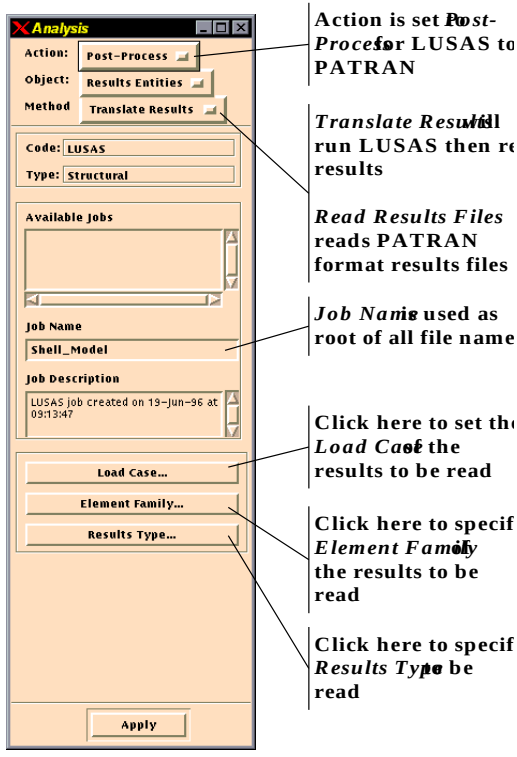
LUSAS to PATRAN

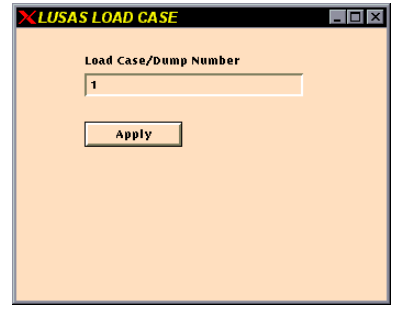
Data is transferred from LUSAS to PATRAN by setting the **Action** options list on the **Analysis** dialog to **Post Proces**s. The Method setting is used as follows:

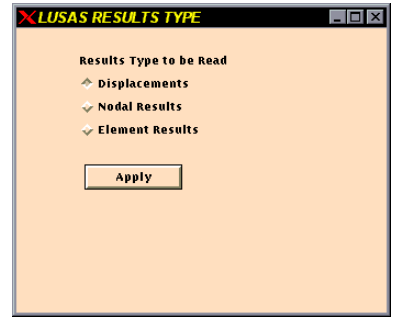
- **Translate Results** LUSPAT will be invoked to generate results files from the LUSAS restart file **jobname.rst**.
- **Read Results Files** it will be assumed that the relevant binary PATRAN results files already exist.

LUSAS to PATRAN Translation Procedure









Analysis Dialog for LUSAS to PATRAN Subordinate dialogs shown right

PATLUS Element Details

The following tables detail which elements are supported by the PATRAN-LUSAS interface and also which loading types are applicable to each element configuration set.

Note. The non-shaded cells in the tables below indicate the elements supported in pre-PATRAN 3 versions of the interface.

BAR 2		Load Types				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/ Elem. Load	Nodal Prescr. Value
BAR2	1	Yes			Yes	Yes
BRS2	2	Yes			Yes	Yes
BEAM	3	Yes	Yes		Yes	Yes
GRIL	4	Yes	Yes		Yes	Yes
BRP2	5	Yes	Yes		Yes	Yes
BMS3	6	Yes	Yes		Yes	Yes
BXM2	7	Yes			Yes	Yes
BXS2	8	Yes				Yes
BFD2	9	Yes				Yes
BFX2	10	Yes				Yes
BFS2	11	Yes				Yes
LFD2	12	Yes				Yes
LFX2	13	Yes				Yes
LFS2	14	Yes				Yes
BTS3	15	Yes				Yes
JNT3	101	Yes				Yes
JPH3	102	Yes				Yes
JF3	103	Yes				Yes

BAR 2		Load Types				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/ Elem. Load	Nodal Prescr. Value
JRP3	104	Yes				Yes
JNT4	105	Yes				Yes
JL43	106	Yes				Yes
JSH4	107	Yes				Yes
JL46	108	Yes				Yes
JSL4	109	Yes				Yes
JAX3	110	Yes				Yes
JXS3	111	Yes				Yes

BAR 3		Load Types				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/ Elem. Load	Nodal Prescr. Value
BAR3	1	Yes			Yes	Yes
BRS3	2	Yes			Yes	Yes
BM3	3	Yes	Yes		Yes	Yes
BSL3	4	Yes	Yes		Yes	Yes
BSL4	5	Yes	Yes		Yes	Yes
BS3	6	Yes	Yes		Yes	Yes
BS4	7	Yes	Yes		Yes	Yes
BMX3	8	Yes	Yes		Yes	Yes
BXS3	9	Yes	Yes		Yes	Yes
BXS4	10	Yes	Yes		Yes	Yes
BXL4	11	Yes	Yes		Yes	Yes

BAR 3		Load Types				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/ Elem. Load	Nodal Prescr. Value
BXM3	12	Yes			Yes	Yes
BXE3	13	Yes				Yes
BFD3	14	Yes				Yes
BFX3	15	Yes				Yes
BFS3	16	Yes				Yes

TRI 3		Load Types				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/ Elem. Load	Nodal Prescr. Value
TPM3	1	Yes		Yes	Yes	Yes
TAX3	2	Yes		Yes	Yes	Yes
TF3	3	Yes	Yes	Yes	Yes	Yes
TRP3	5	Yes	Yes	Yes	Yes	Yes
TSM3	7	Yes	Yes	Yes	Yes	Yes
TS3	9	Yes	Yes	Yes	Yes	Yes
TFD3	13	Yes		Yes	Yes	Yes
TXF3	14	Yes		Yes	Yes	Yes
TPN3	16	Yes		Yes	Yes	Yes
TPM3E	17	Yes		Yes	Yes	Yes
TAX3E	18	Yes		Yes	Yes	Yes
TPN3E	19	Yes		Yes	Yes	Yes
TAX3F	20	Yes		Yes	Yes	Yes
TTS3	21	Yes		Yes	Yes	Yes

TRI 6		Load Types				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/ Elem. Load	Nodal Prescr. Value
TPM6	1	Yes		Yes	Yes	Yes
TAX6	2	Yes		Yes	Yes	Yes
TF6	3	Yes		Yes	Yes	Yes
TTF6	4	Yes		Yes	Yes	Yes
TSL6	5	Yes		Yes	Yes	Yes
TFD6	6	Yes		Yes	Yes	Yes
TXF6	7	Yes		Yes	Yes	Yes
TPK6	8	Yes		Yes	Yes	Yes
TXK6	9	Yes		Yes	Yes	Yes
TPN6	16	Yes		Yes	Yes	Yes
TNK6	17	Yes		Yes	Yes	Yes
TAX6F	18	Yes		Yes	Yes	Yes
TTS6	19	Yes		Yes	Yes	Yes

QUAD 4		Load Types				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/ Elem. Load	Nodal Prescr. Value
QPM4	1	Yes		Yes	Yes	Yes
QAX4	2	Yes		Yes	Yes	Yes
QF4	3	Yes	Yes	Yes	Yes	Yes
QSC4	4	Yes	Yes	Yes	Yes	Yes
QRP4	5	Yes	Yes	Yes	Yes	Yes
RPI4	6	Yes	Yes	Yes	Yes	Yes

QUAD 4		Load Types				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/ Elem. Load	Nodal Prescr. Value
QSM4	7	Yes	Yes	Yes	Yes	Yes
SMI4	8	Yes	Yes	Yes	Yes	Yes
QS4	9	Yes	Yes	Yes	Yes	Yes
QSH4	10	Yes	Yes	Yes	Yes	Yes
SHI4	11	Yes	Yes	Yes	Yes	Yes
QSI4	12	Yes	Yes	Yes	Yes	Yes
QFD4	13	Yes		Yes	Yes	Yes
QXF4	14	Yes		Yes	Yes	Yes
PMI4	15	Yes		Yes	Yes	Yes
QPN4	16	Yes		Yes	Yes	Yes
QPM4E	17	Yes		Yes	Yes	Yes
QAX4E	18	Yes		Yes	Yes	Yes
QPN4E	19	Yes		Yes	Yes	Yes
QAX4F	20	Yes		Yes	Yes	Yes
QTS4	21	Yes		Yes	Yes	Yes
QPM4M	22	Yes		Yes	Yes	Yes
QPN4M	23	Yes		Yes	Yes	Yes
QAX4M	24	Yes		Yes	Yes	Yes

QUAD 8		Load Types				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/ Elem. Load	Nodal Prescr. Value
QPM8	1	Yes		Yes	Yes	Yes

QUAD 8		Load Types				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/ Elem. Load	Nodal Prescr. Value
QAX8	2	Yes		Yes	Yes	Yes
QF8	3	Yes	Yes	Yes	Yes	Yes
QTF8	4	Yes	Yes	Yes	Yes	Yes
QSL8	5	Yes	Yes	Yes	Yes	Yes
QFD8	6	Yes		Yes	Yes	Yes
QXF8	7	Yes		Yes	Yes	Yes
QPK8	8	Yes		Yes	Yes	Yes
QXK8	9	Yes		Yes	Yes	Yes
QPN8	16	Yes		Yes	Yes	Yes
QNK8	17	Yes		Yes	Yes	Yes
QAX8F	18	Yes		Yes	Yes	Yes
QTS8	19	Yes		Yes	Yes	Yes

QUAD 12		Load Types				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/ Elem. Load	Nodal Prescr. Value
QP12	1	Yes		Yes	Yes	Yes
QA12	2	Yes		Yes	Yes	Yes

TET 4		Load Types				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/ Elem. Load	Nodal Prescr. Value
TH4	1	Yes				
TF4	2	Yes		Yes	Yes	Yes
TH4E	3	Yes		Yes	Yes	Yes

TET 10		Load Types				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/ Elem. Load	Nodal Prescr. Value
TH10	1	Yes		Yes	Yes	Yes
TF10	2	Yes		Yes	Yes	Yes

WEDG 6		Load Types				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/ Elem. Load	Nodal Prescr. Value
PN6	1	Yes		Yes	Yes	Yes
PF6	2	Yes		Yes	Yes	Yes
PN6E	3	Yes		Yes	Yes	Yes

WEDG 12		Load Types				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/ Elem. Load	Nodal Prescr. Value
PN12	1	Yes		Yes	Yes	Yes
PF12	2	Yes		Yes	Yes	Yes

WEDG 15		Load Types				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/ Elem. Load	Nodal Prescr. Value
PN15	1	Yes		Yes	Yes	Yes
PF15	2	Yes		Yes	Yes	Yes

HEX 8		Load Types				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/ Elem. Load	Nodal Prescr. Value
HX8	1	Yes		Yes	Yes	Yes
HF8	2	Yes		Yes	Yes	Yes
HX8E	3	Yes		Yes	Yes	Yes
HX8M	4	Yes		Yes	Yes	Yes

HEX 16		Load Types				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/ Elem. Load	Nodal Prescr. Value
HX16	1	Yes		Yes	Yes	Yes
HF16	2	Yes		Yes	Yes	Yes

HEX 20		Load Types				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/ Elem. Load	Nodal Prescr. Value
HX20	1	Yes		Yes	Yes	Yes
HF20	2	Yes		Yes	Yes	Yes

LUSPAT Results Columns

Displacement Results

The displacement results files have the following column definitions. Displacements are in global directions unless transformed directions have been selected.

X, Y, Z	Translations	**THX,THY,THZ**	Rotations	**T**	Field value

Col.	Value	Col.	Value
1	X translation or T nodal temperature	**4**	THX rotation
2	Y translation	**5**	THY rotation
3	Z translation	**6**	THZ rotation

Average Nodal and Element Results

The averaged nodal results and element results have the following column definitions. Averaged nodal results are in global directions unless transformed directions have been selected.

SX, SY, SZ	Direct stresses	**SE**	Von Mises equivalent stress
SXY, SYZ, SXZ	Shear stresses	**S1, S2, S3**	Principal stresses

Nodal/Element Results Beam

Not supported.

Nodal/Element Results Plane Membrane

Col.	Value	Col.	Value
1	Stress in X direction (SX)	**4**	Von Mises stress (SE)
2	Stress in Y direction (SY)	**5**	Maximum principal stress (S1)
3	Stress in XY direction (SXY)	**6**	Minimum principal stress (S2)

Nodal/Element Results Space Membrane

Col.	Value	Col.	Value
1	Stress in X direction (SX)	6	Stress in ZX direction (SZX)
2	Stress in Y direction (SY)	7	Von Mises stress (SE)
3	Stress in Z direction (SZ)	8	Maximum principal stress (S1)
4	Stress in XY direction (SXY)	9	Principal stress (S2)
5	Stress in YZ direction (SYZ)	10	Minimum principal stress (S3)

Nodal/Element Results Plate Flexure

Col.	Value	Surface	Col.	Value	Surface
1	Stress in X direction (SX)	Top	7	Stress in X direction (SX)	Bottom
2	Stress in Y direction (SY)	Top	8	Stress in Y direction (SY)	Bottom
3	Stress in XY direction (SXY)	Top	9	Stress in XY direction (SXY)	Bottom
4	Von Mises stress (SE)	Top	10	Von Mises stress (SE)	Bottom
5	Max principal stress (S1)	Top	11	Max principal stress (S1)	Bottom
6	Min principal stress (S2)	Top	12	Min principal stress (S2)	Bottom

Nodal/Element Results Shells

Col.	Value	Surface	Col.	Value	Surface
1	Stress in X direction (SX)	Top	11	Stress in X-direction (SX)	Bottom
2	Stress in Y direction (SY)	Top	12	Stress in Y-direction (SY)	Bottom
3	Stress in Z direction (SZ)	Top	13	Stress in Z-direction (SZ)	Bottom
4	Stress in XY direction (SXY)	Top	14	Stress in XY direction (SXY)	Bottom
5	Stress in YZ direction (SYZ)	Top	15	Stress in YZ direction (SYZ)	Bottom
6	Stress in ZX direction (SZX)	Top	16	Stress in ZX direction (SZX)	Bottom
7	Von Mises stress (SE)	Top	17	Von Mises stress (SE)	Bottom
8	Max principal stress (S1)	Top	18	Max principal stress (S1)	Bottom
9	Principal stress (S2)	Top	19	Principal stress (S2)	Bottom
10	Min principal stress (S3)	Top	20	Min principal stress (S3)	Bottom

Nodal/Element Results Axisymmetric Solid/Plane Strain

Col.	Value	Col.	Value
1	Stress in X direction (SX)	5	Von Mises stress (SE)
2	Stress in Y direction (SY)	6	Maximum principal stress (S1)
3	Stress in XY direction (SXY)	7	Minimum principal stress (S2)
4	Hoop stress		

Nodal/Element Results Solids

Col.	Value	Col.	Value
1	Stress in X-direction (SX)	6	SZX Stress in ZX-direction
2	Stress in Y-direction (SY)	7	SE Von Mises stress
3	Stress in Z-direction (SZ)	8	S1 Maximum principal stress
4	SXY Stress in XY-direction	9	S2 Principal stress
5	SYZ Stress in YZ-direction	10	S3 Minimum principal stress

Nodal/Element Results Field (1D)

Not supported.

Nodal/Element Results Field (2D)

Col.	Value
1	dT(X) Gradient in X direction
2	dT(Y) Gradient in Y direction

Nodal/Element Results Field (3D)

Col.	Value
1	dT(X) Gradient in X direction
2	dT(Y) Gradient in Y direction
3	dT(Z) Gradient in Z direction

System Parameters

When using the LUSAS to PATRAN interface default responses may be overridden with system parameters by specifying values in a start-up file (**patlus.str**/**luspat.str**). The start-up file should be created in the working directory and should be formatted as in the example shown below:

```
SYSTEM
ISTDEF=1
IELSTR=1
EXIT
```

The following system parameters are supported:

Parameter	Default	Description
INTFP3	0	PATRAN Version identifier (0 for P2.5, 1 for P3).
IASCII	1	Format for output of results files. (0 for binary). The results files default to an ASCII format but may be selected as binary by specifying the system parameter IASCII=0 (this will be carried out automatically when running LUSPAT from within PATRAN). ASCII PATRAN results files are easily transferable between different machine types and can be modified to change the title information if required. Binary files require less disc storage and can be read faster by PATRAN but cannot be modified directly. The READER utility supplied with PATRAN permits conversion between ASCII and binary formats if required.
IBATCH	0	Switch to allow LUSPAT to be run in batch with all prompting suppressed (1 for batch operation).
ISLIDE	1	Switch to assume that when run in batch, face loads with the prescribed value should be converted to slides (0 for no conversion).
NDUMP	0	Restart dump number selected.
NSTRUT	0	Structure type number for stress averaging.
NMAT	0	Material group number for stress averaging.
IEIGNL	0	Switch to add eigenvectors to displacements for nonlinear problems. (1 adds eigenvectors to nonlinear displacements)
ISTDEF	0	Switch to assemble PATRAN definition file for elements topology and node coordinates (1 creates **.def** file). The structure topology and coordinates are not translated into PATRAN format by default but may be required when a model not generated by PATRAN is to be processed subsequently. To generate a PATRAN definition file set the system parameter ISTDEF=1. The node and element records of structure definition will be output in a file with the **.def** extension. Transformed freedoms are not supported.
INDDSP	1	Assemble displacement file (0 suppresses displacement results). The displacements or temperatures (field values) are translated by default. To suppress their creation set the

Parameter	Default	Description
		system parameter INDDSP=0.
INDSTR	1	Switch to assemble averaged nodal results (0 suppresses averaged nodal results). The averaged nodal results for the selected structure type are translated by default. To suppress their creation set the system parameter INDSTR=0. The column definitions for the average nodal results are described in tables starting on page *27*.
IELSTR	0	Switch to assemble element centroid results (1 outputs element centroid results). The values at element centroid positions for the selected structure are not translated by default. To translate element values set the system parameter IELSTR=1. The column definitions for the element results are described on page 27.
IRSTRF	0	Switch to output results in transformed directions (1 outputs results in transformed directions)
ICNGDF	0	Default element configuration code to use when zero configuration code encountered. If not specified PATLUS will prompt for an element name when a zero configuration code is encountered.

FEMView and FAM Interface

Introduction

This section describes the use of FVLUSAS, the LUSAS Solver to FEMView/FAM export program supported by FEA Ltd.

The program reads results from the LUSAS restart file. The LUSAS restart file is named **jobname.rst.** A restart file is created by LUSAS using the RESTART WRITE or RESTART WRITE BRIEF commands in the data file or using the appropriate OUTPUT option in various analysis control sections in the data file.

FVLUSAS creates a sequential ASCII file named **model.prv**.

LUSAS to FEMView/FAM

Running The Program

The interface is designed to be interactive and hence should be initiated from a command prompt. Before data processing begins the interface prompts for certain information allowing you to select your output requirements. The prompts are outlined in the next few paragraphs:

```
Enter FEMVIEW model name (up to 6 characters)_:
```

Enter the dataset name. The default model name is the current jobname.

```
Is this a new FEMVIEW model [Y]_:
```

Identify the data set as a new PREVIEW data set or, if you wishes to append additional information to an existing PREVIEW model, an existing data set.

```
Input dump number required (zero for all dumps) [0]_:
```

By default all the restart dumps will be processed. In linear runs all the results are stored in restart dump 1. For nonlinear or dynamic analysis the restart dump numbers are as specified by the user in the analysis control data.

The remaining options refer to the type of information to be transferred to the Preview data file. All results are not available for all element types. Consult the section titled *FEMView/FAM Results Availability* on page *36* for full details.

Generation of Load Case Names

FEMView identifies each load case by a six character name. The interface will generate these load case names automatically and you have no control over the chosen names.

- **Linear** For linear problems, load cases will be named LC0001, LC0002 etc.
- **Eigenvalue** For eigenvalue problems, eigenvectors will be named LC0001, LC0002 etc.
- **Nonlinear** For nonlinear transient field and step-by-step dynamic problems only one load case per restart dump is written. For such problems the result dumps are referred to as D01L01, D02L01 etc. where D refers to the dump number, e.g. D01 refers to dump 1, and L refers to the load case number within that dump.

Average Stress Results Grouping

Since it is invalid to average stress results across material discontinuities, average stress results are available by material groups only. These groups are defined by the material assignment numbers in the LUSAS data deck.

Supported Elements

For convenience in specifying the available results supported LUSAS elements have been divided into groups. These groups are shown in the following table.

No	Group Name	LUSAS Elements
1	Solids	HX8, HX16, HX20, PN6, PN12, PN15, TH4, TH10, HX8M, HX8E, PN6E, TH4E
2	Space Membrane	TSM3, QSM4, SMI4
3	Plate Membrane	TPM3, TPM6, QPM4, QPM8, TPM3E, QPM4E

No	Group Name	LUSAS Elements
4	Axisymmetric Solid/Plane Strain	TAX3, TAX6, QAX4, QAX8, TAX3E, QAX4E, TPN3, TPN6, QPN4, QPN8, TPN3E, QPN4E
5	Thick Plate Flexure	TTF6, QTF8, QSC4, QSC8
6	Thin Plate Flexure	TF3, TF6, QF4, QF8
7	Ribbed Plate	TRP3, QRP4, RPI4
8	Semi-loof Shell	TSL6, QSL8
9	Flat Shell	QSH4, QSH6, SHI4, SHI6
10	Bar	BAR2, BAR3, BRS2, BRS3
11	2D Beam Explicit Integration	BEAM
12	Grillage Explicit Integration	GRIL
13	Ribbed Beam Explicit Integration	BRP2
14	3D Beam Explicit Integration	BMS3
15	2D Thin Beam	BM3
16	3D Thin Beam	BS3, BS4, BXS4, BSL3, BSL4, BXL4
17	3D Thick Beam	BTS4, BTS6, BTW6, BOX6
18	Joint	JNT4, JNT3, JPH3, JAX3, JF3, JRP3, JSL4, JSH4, JXS3, JL43, JL46, LFD2, LFX2, LFS2
19	2D Field	TFD3, TFD6, QFD4, QFD8, TXF6, QXF4, QXF8, QXF9
20	Axisymmetric Sheet	BXM2, BXM3
21	Axisymmetric Shell	BXS3, BXE3
22	3D Field	HF8, HF16, HF20, PF6, PF12, PF15, TF4, TF10
23	1D Field	BFD2, BFD3, BFX2, BFX3, BFS2, BFS3
24	Fourier Elements	TAX3F, TAX6F, QAX4F, QAX8F

No	Group Name	LUSAS Elements
25	Thick Shell Elements	TTS3, QTS4, TTS6, QTS8

The group numbers are broadly dependent on the stress/stress resultant output produced from the analysis and determine which results data is available.

FEMView/FAM Results Availability

No	Nodal Disp.	Nodal Temp.	Avge. Nodal Stress	Nodal Prin. Stress	Nodal von Mises Stress	Elem. General Stress	Elem. Stress
1	Yes	n/a	Yes	Yes	Yes	Yes	Yes
2	Yes	n/a	Yes	Yes	Yes	Yes	Yes
3	Yes	n/a	Yes	Yes	Yes	Yes	Yes
4	Yes	n/a	Yes	Yes	Yes	n/a	Yes
5	Yes	n/a	Yes	Yes	Yes	Yes	Yes
6	Yes	n/a	Yes	Yes	Yes	Yes	Yes
7	Yes	n/a	Yes	Yes	Yes	Yes	Yes
8	Yes	n/a	Yes	Yes	Yes	Yes	Yes
9	Yes	n/a	Yes	Yes	Yes	Yes	Yes
10	Yes	n/a	n/a	n/a	n/a	Yes	n/a
11	Yes	n/a	n/a	n/a	n/a	Yes	n/a
12	Yes	n/a	n/a	n/a	n/a	Yes	n/a
13	Yes	n/a	n/a	n/a	n/a	Yes	n/a
14	Yes	n/a	n/a	n/a	n/a	Yes	n/a
15	Yes	n/a	n/a	n/a	n/a	Yes	n/a
16	Yes	n/a	n/a	n/a	n/a	Yes	n/a
17	Yes	n/a	n/a	n/a	n/a	Yes	n/a
18	Yes	Yes	n/a	n/a	n/a	n/a	n/a
19	n/a	Yes	Yes	Yes	Yes	n/a	Yes

No	Nodal Disp.	Nodal Temp.	Avge. Nodal Stress	Nodal Prin. Stress	Nodal von Mises Stress	Elem. General Stress	Elem. Stress
20	**Yes**	n/a	n/a	n/a	n/a	n/a	**Yes**
21	**Yes**	n/a	n/a	n/a	n/a	n/a	**Yes**
22	n/a	**Yes**	**Yes**	**Yes**	**Yes**	n/a	**Yes**
23	n/a	**Yes**	n/a	n/a	n/a	n/a	n/a
24	n/a	n/a	n/a	n/a	n/a	n/a	n/a
25	**Yes**	n/a	**Yes**	**Yes**	**Yes**	**Yes**	**Yes**

Notes

- Since joint elements are not supported by FEMView any results for joints will be ignored.
- Displacements are always output in the system global axis set.
- Element general stresses are available for all beam, plate and shell elements and some other elements. For these elements these stresses refer to the local element axis set, which is determined by the node order as specified in the LUSAS element topology. For shells and plate flexure elements the stresses at the mid-surface and bending moments are available.
- Element stress resultants are available for all but the 3D and axisymmetric solid elements.

FEMView/FAM Stress Types

No	Stress Components						No	Stress Components					
1	σX	σY	σZ	σXY	σYZ	σXZ	**14**	Fx	Fy	Fz	Mx	My	Mz
2	Nx	Ny	Nxy				**15**	Fx	Mz				
3	sX	sY	sXY				**16**	Fx	Mx	My	Mz		
4	sX	sY	sZ	sXY			**17**	Fx	Fy	Fz	Mx	My	Mz
5	MX	MY	MXY	SX	SY		**18**	-					
6	MX	MY	MXY				**19**	gX	gY	gXY			

No	Stress Components						No	Stress Components					
7	Nx	Ny	Nxy	Mx	My	Mxy	**20**	Fx					
8	Nx	Ny	Nxy	Mx	My	Mxy	**21**	Fx	Mz				
9	Nx	Ny	Nxy	Mx	My	Mxy	**22**	gX	gY	gZ	gXY	gYZ	gZX
10	Fx						**23**	gx					
11	Fx	Fy	Mz				**24**	-					
12	Fx	Mx	My				**25**	Nx	Ny	Nxy	Mx	My	Mxy
13	Fx	Fy	Fz	Mx	My								

Table Key

F	Force	**σ**	Stress
N	Force Resultant	**g**	Field Gradient
M	Moment	**X, Y, Z**	Global directions are indicated by capital letters
S	Shear Force	**x, y, z**	Local direction results are indicated by lower case letters